

[FreeCAD Forum >](#)

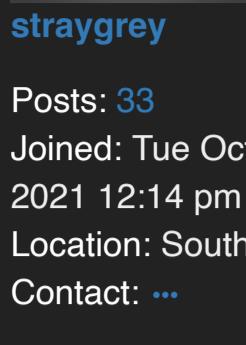
Openscad to cnc mill

Forum rules

Be nice to others! Respect the FreeCAD code of conduct!

Search this topic...

First unread post • 10 posts • Page 1 of 1



straygrey

Posts: 33
Joined: Tue Oct 05, 2021 12:14 pm
Location: South Africa
Contact: ...

Openscad to cnc mill

by **straygrey** » Sun Apr 16, 2023 1:01 pm

Being a long time programmer I find openscad a convenient editing platform and have therefore created

[DavidPhotoBoxCNC.scad](#)

(735 Bytes) Downloaded 76 times

I have successfully 3d printed this file but now wish to cut it from a piece of wood. So far I have

[DavidPhotoBlockCNC.FCStd](#)

(35.54 kB) Downloaded 79 times

but I am somewhat lost and do not know what my next step should be.

Please advise.

OS: Debian GNU/Linux 11 (bullseye) (GNOME/gnome)

Locale: English/South Africa (en_ZA)



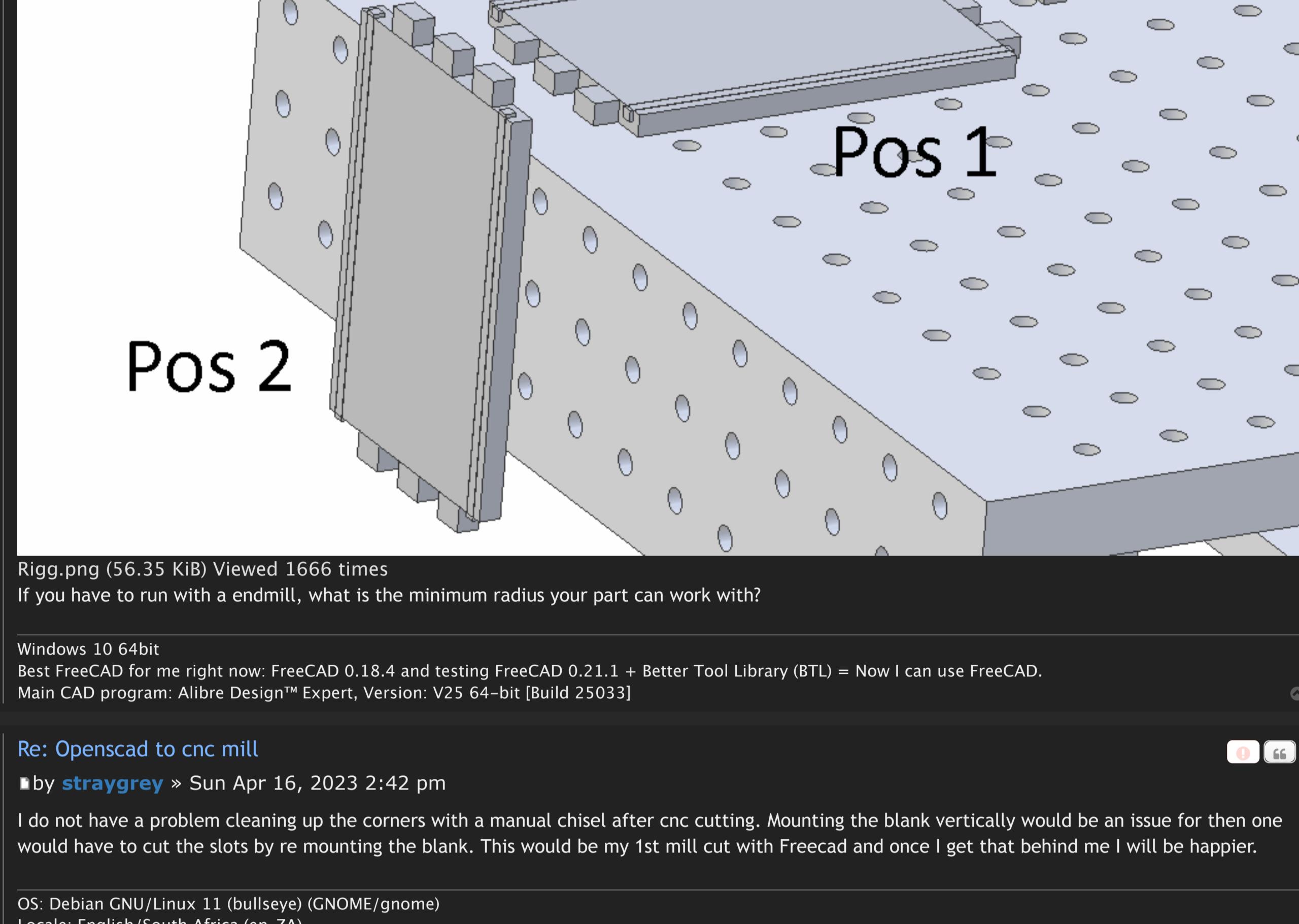
lgrfbs

Posts: 142
Joined: Wed Apr 07, 2021 9:51 am
Location: Sweden
Contact: ...

Re: Openscad to cnc mill

by **lgrfbs** » Sun Apr 16, 2023 2:11 pm

If you are going to have sharp edges, can you mount your blank like this in the machine?



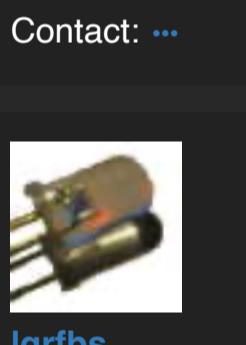
Rigg.png (56.35 KiB) Viewed 1666 times

If you have to run with an endmill, what is the minimum radius your part can work with?

Windows 10 64bit

Best FreeCAD for me right now: FreeCAD 0.18.4 and testing FreeCAD 0.21.1 + Better Tool Library (BTL) = Now I can use FreeCAD.

Main CAD program: Alibre Design™ Expert, Version: V25 64-bit [Build 25033]



straygrey

Posts: 33
Joined: Tue Oct 05, 2021 12:14 pm
Location: South Africa
Contact: ...

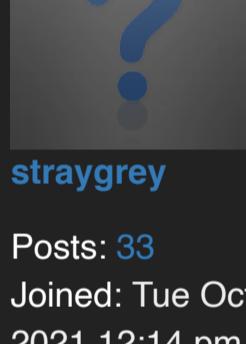
Re: Openscad to cnc mill

by **straygrey** » Sun Apr 16, 2023 2:42 pm

I do not have a problem cleaning up the corners with a manual chisel after cnc cutting. Mounting the blank vertically would be an issue for then one would have to cut the slots by re mounting the blank. This would be my 1st mill cut with Freecad and once I get that behind me I will be happier.

OS: Debian GNU/Linux 11 (bullseye) (GNOME/gnome)

Locale: English/South Africa (en_ZA)



lgrfbs

Posts: 142
Joined: Wed Apr 07, 2021 9:51 am
Location: Sweden
Contact: ...

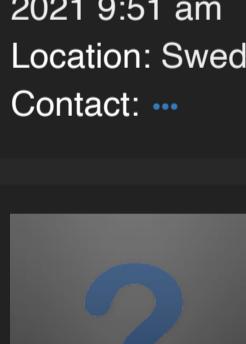
Re: Openscad to cnc mill

by **lgrfbs** » Sun Apr 16, 2023 2:47 pm

Of course I could also do a dogbone. Will look at that when I learn more.

OS: Debian GNU/Linux 11 (bullseye) (GNOME/gnome)

Locale: English/South Africa (en_ZA)



straygrey

Posts: 33
Joined: Tue Oct 05, 2021 12:14 pm
Location: South Africa
Contact: ...

Re: Openscad to cnc mill

by **straygrey** » Sun Apr 16, 2023 3:20 pm

Yes I do have a 3018 machine and tools.

I perceive that I have a number of Jobs to undertake as follows

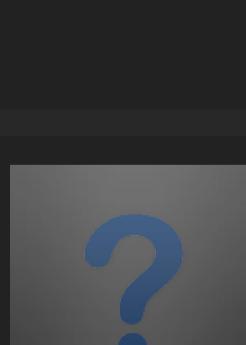
!) A job to cut each of the 3 slots. ie 1 job per slot

2) A job to cut the complete side out of the base.

My first issue is that when I select the 1st slot by clicking on it in the Combo view Model and expect to be able to create a job by clicking on the JobCreate icon but cannot as the job create icon is greyed out. What am I misunderstanding or what did I do incorrectly?

OS: Debian GNU/Linux 11 (bullseye) (GNOME/gnome)

Locale: English/South Africa (en_ZA)



lgrfbs

Posts: 142
Joined: Wed Apr 07, 2021 9:51 am
Location: Sweden
Contact: ...

Re: Openscad to cnc mill

by **lgrfbs** » Sun Apr 16, 2023 4:07 pm

Which version of FreeCAD do you have?

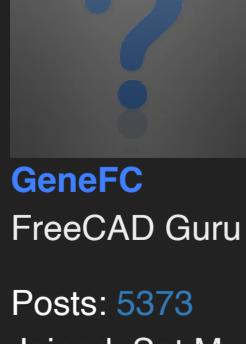
I don't have much more to say, so you'll have to rely on the other professionals on this forum.

Just because I am curious which tool will you use?

Windows 10 64bit

Best FreeCAD for me right now: FreeCAD 0.18.4 and testing FreeCAD 0.21.1 + Better Tool Library (BTL) = Now I can use FreeCAD.

Main CAD program: Alibre Design™ Expert, Version: V25 64-bit [Build 25033]



straygrey

Posts: 33
Joined: Tue Oct 05, 2021 12:14 pm
Location: South Africa
Contact: ...

Re: Openscad to cnc mill

by **straygrey** » Sun Apr 16, 2023 4:55 pm

See my signature. I know that I am not supposed to put the Freecad version in the signature but until I learn a better way this will have to do.

Code: Select all

OS: Debian GNU/Linux 11 (bullseye) (GNOME/gnome)

Word size of FreeCAD: 64-bit

Version: 0.20.1.29410 (Git) AppImage

Build type: Release

Branch: (HEAD detached at 0.20.1)

Hash: f5d13554ecc70456fb6e970568ae5c74ba727563

Python 3.10.5, Qt 5.15.4, Coin 4.0.0, Vtk 9.1.0, OCC 7.6.2

Locale: English/South Africa (en_ZA)

Installed mods:

* Help 1.0.3

OS: Debian GNU/Linux 11 (bullseye) (GNOME/gnome)

Locale: English/South Africa (en_ZA)

[FreeCAD Forum >](#)